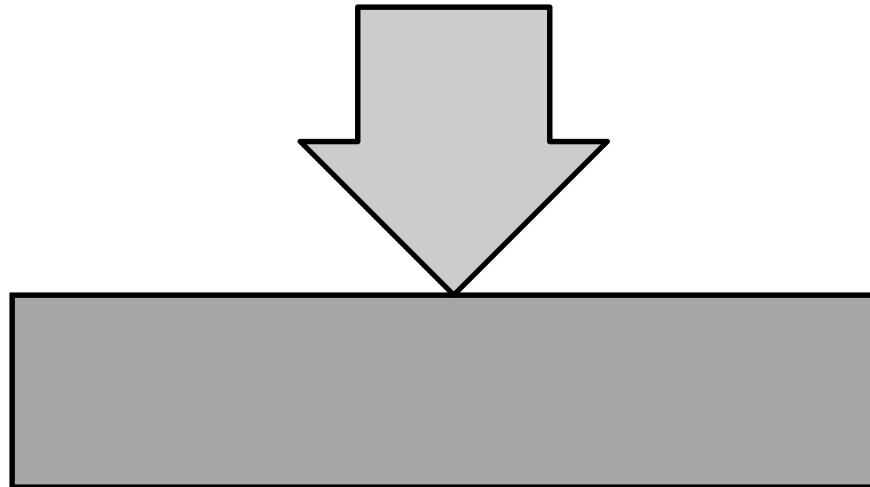# EverFE v1.01 User's Manual

A joint project of the
University of Washington
and the
Washington State Department of Transportation

EVERFE V1.01 USER'S MANUAL

1. Overview

EverFE is a rigid pavement analysis tool built around cutting-edge finite element technology. **EverFE** allows the user to specify all the parameters of the problem interactively, with immediate visual feedback. Once the parameters have been set and the solver run on a problem, **EverFE** allows the user to view the deformations and stresses, both graphically - as wireframes and stress colormaps - and numerically.

The general method for using **EverFE** may be summarized as follows:

- Specify dimensions, material properties, and loads
- Specify degree of mesh refinement and run the solver
- View the results to determine if they fall within acceptable parameters

When started, the **EverFE** screen looks like this:
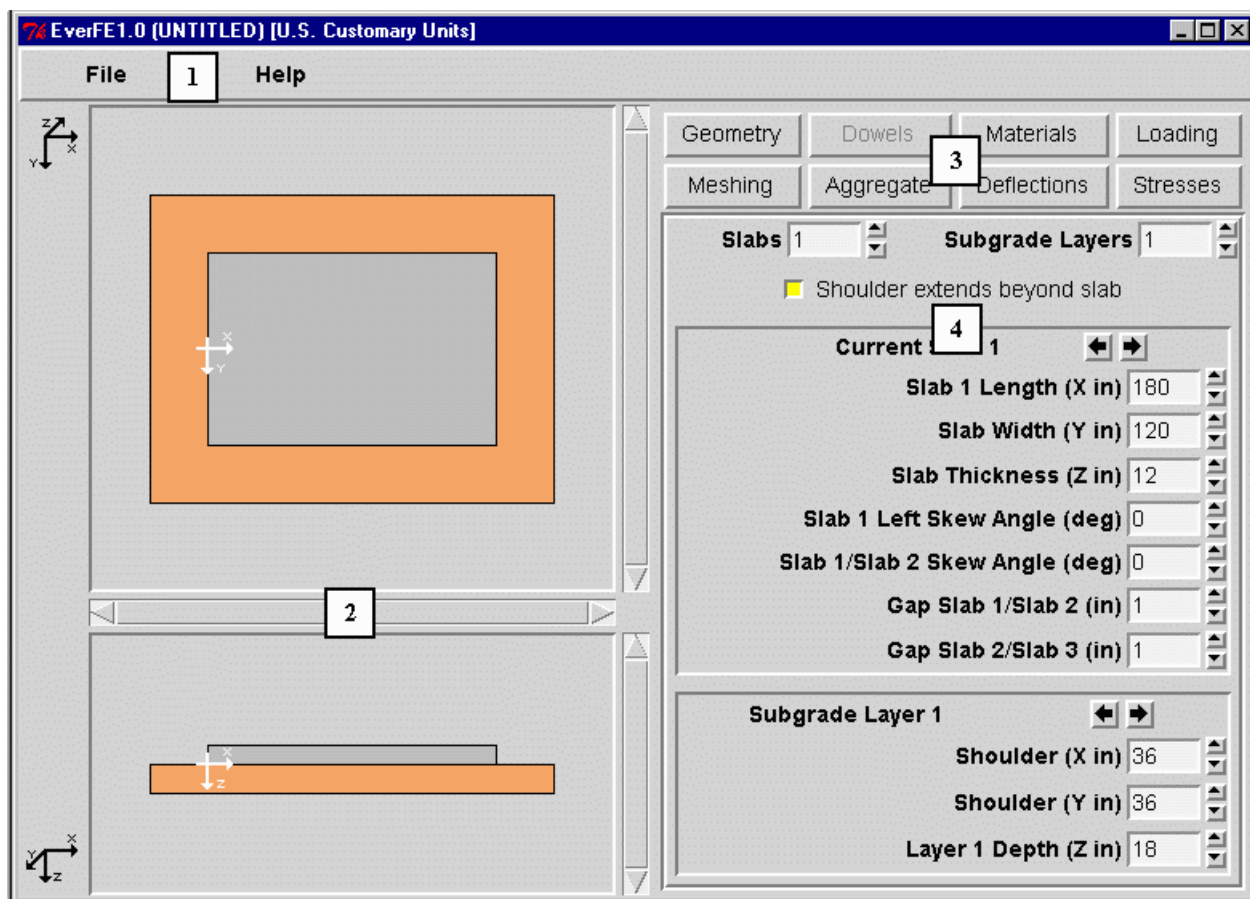
The main **EverFE** window consists of several parts:

1) The **Menu Bar**, providing access to the *File* and *Help* menus

2) The **Plan** and **Elevation** views of the current problem. Note the coordinate system: **X** is along the direction of travel; **Y** is transverse; and **Z** is vertical, with the positive **Z** axis pointing *downwards*. The origin of the coordinate system is centered on the leftmost side of the first slab, with **Z** = 0 at the bottom of the slab - i.e, the interface between the slab and the subgrade.

3) The **Button Bar** that pulls up the various control panels

4) The **Control Panel** area, where the controls appear

These subjects are discussed in more detail in their respective sections, but in brief:

The *File* menu provides for opening and saving Projects.

The plan and elevation views provide instant graphic feedback of the current problem geometry.

The button bar governs which control panel is currently displayed; pressing the "Dowels" button, for instance, will bring up the Dowel control panel.

The control panels provide control over the various aspects of an **EverFE** problem. The first five panels - Geometry, Dowels, Materials, Loading, and Aggregate - can be thought of as the "pre-processor" portion of the program: with these panels, the user specifies the physical parameters of the project.

- The Geometry panel specifies the number and dimensions of the slabs and the subgrade layers. There may be from one to three slabs, and from zero to three subgrade layers. The shoulder - the portion of the subgrade extending horizontally beyond the slabs - may be explicitly modeled as well.

- The Dowels panel controls the number and placement of the dowels, the steel bars that transfer shear between slabs. There are two primary options for dowel placement: *even* spacing, and *wheelpath* spacing; in addition, it is possible to specify the location of each dowel individually. The embedment of the dowel may also be specified, as may gaps between the dowels and the slabs.
  [Note: this panel is not available unless there are two or more slabs.]

- The Materials panel sets the material properties of the various components of the project - the slabs, the subgrade, the dense liquid foundation, and the dowels.

- The Loading panel allows the user to place loads on the roadbed. For convenience, there are several types of loads: point, circular patch, rectangular patch, and axle (i.e, dual loads of the given type.)

- The Aggregate panel controls whether aggregate interlock will be modeled, and what the parameters are.

If the previous panels were the "pre-processor," then the next one is the "processor" itself.

- The Meshing panel is the heart of the program - here, the user can set the finite element mesh parameters and run the FE solver. Once the solver has been run, a number of data files are written into the project directory, and the results can be viewed, either immediately or at a later time.

The last two panels constitute the "post-processor." The FE solver generates displacement and stress data at every node in the mesh; examining this data by hand is impractical, so **EverFE** displays it visually.

- The Deflections panel enables visualization of the deformation of the slabs and the subgrade.
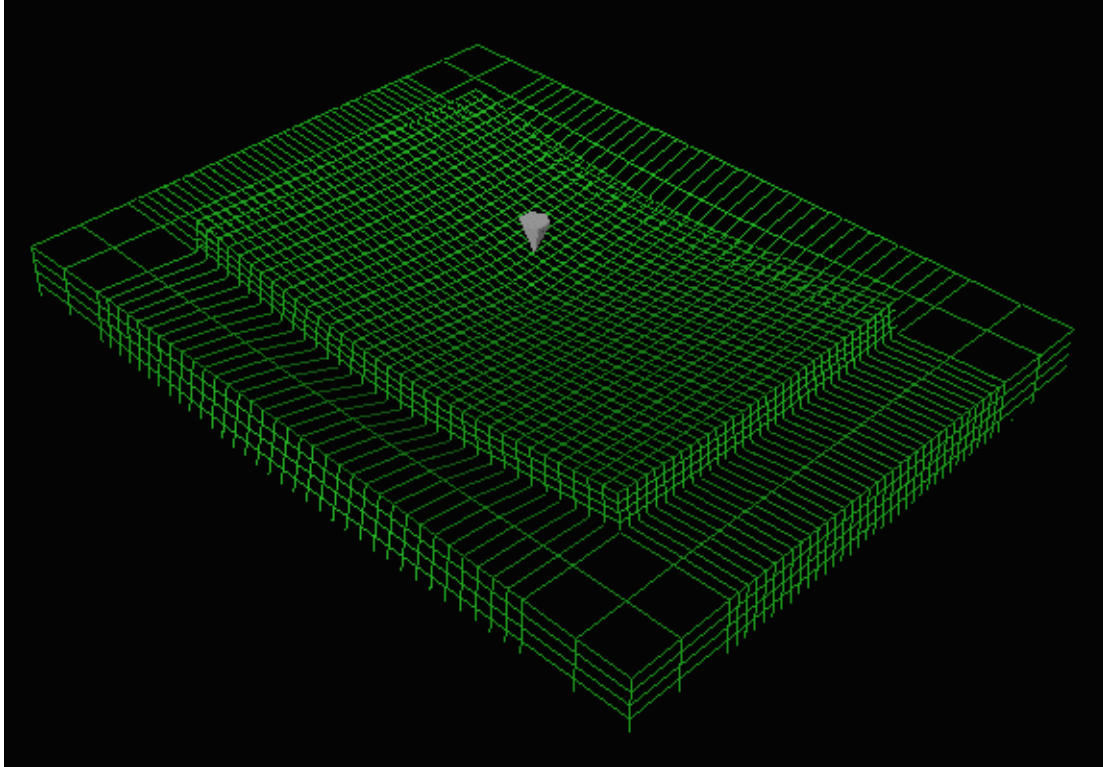
A sample deformation view

- The <u>Stresses</u> panel enables visualization of the solution stresses. It is also possible to get direct numerical data for a given point from this panel.
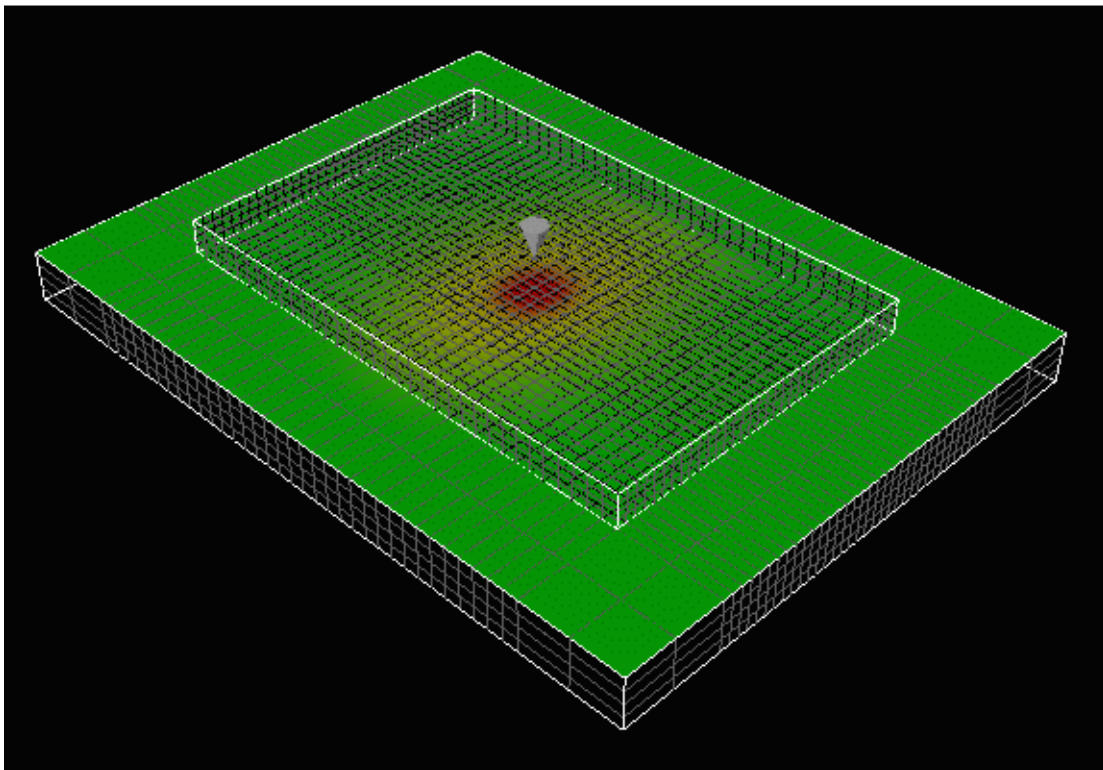
A sample stress view

2. Projects and the File Menu

The **File** menu allows the user to create, save, and open **EverFE** projects. A project consists of the user-specified geometry and loads for a particular problem, and - if the solver has been run - a set of data files containing stress and deformation data.

File | New create a new project with default values, and allows the user to select either metric or English units. Note that the unit system is fixed at the start of model development, and may not be changed thereafter.

File | Open opens an existing project.

File | Save saves the current geometry. Note that there is no need to save the finite element data; that is automatically written to disk after the solver has run.

File | Save As saves under a user-specified name.

File | Exit exits **EverFE**.

3. Panel Descriptions

3.1 Geometry Panel

3.2

The **Geometry** panel governs the number and dimensions of the slabs and subgrade layers involved in the problem.

The **Slabs** control sets the number of rigid pavement slabs modeled - either one, two, or three.

The **Subgrade Layers** control sets how many layers of subgrade material there will be - either zero, one, two, or three.

Having multiple subgrade layers allows the user to specify different material properties for each layer; see the *Materials* panel.

The **Shoulder Extends Beyond Slab** checkbox governs whether or not the subgrade has a shoulder extension beyond the edges of the slab.

The **Current Slab** arrow buttons adjust which slab's controls are currently displayed. Some of the dimensions are universal to all the slabs - for instance, setting the "Slab Width" dimension affects all slabs displayed - but some are specific to individual slabs or slab ends. The length (**X** dimension) of each slab can be set independently, as can the skew angles at each slab end or joint.

The **Slab # Length (X in)** control sets the length of the current slab, in inches (all measurements are in English units.) Length is measured along the centerline (**Y** = 0) of the slab.

The **Slab Width (Y in)** control sets the transverse dimension of *all* slabs.

The **Slab Thickness (Z in)** control sets the thickness of *all* slabs.

The **Skew Angle** controls adjust the angle by which each slab end is skewed from the orthogonal. The ends rotate about the centerline of the slab (i.e, **Y** = 0) and angles are measured in degrees positive *clockwise*. There are four possible skew angles that can be set - in the case of a three-slab problem - and the controls affecting the *left* and *right* ends of the current slab are displayed at any given time. The four possible skew angles are:

- **Slab 1 Left Skew Angle**
- **Slab 1/Slab 2 Skew Angle**
- **Slab 2/Slab 3 Skew Angle**
- **Slab 3 Right Skew Angle**

The **Gap Slab 1/Slab 2** and **Gap Slab 2/Slab 3** controls govern the joint distance between two slabs. Note that the dimension is measured *perpendicular* to the edge of the slabs. Also note that this dimension significantly affects the aggregate interlock shear transfer when the nonlinear aggregate interlock model is used.

The **Subgrade Layer #** arrow buttons control which layer of subgrade material is currently selected - the right arrow moves to the next lower subgrade layer; the left arrow moves to the next higher. You can also select a subgrade layer by clicking on it in the elevation view.
[N.B.: This is also how you bring up the material properties for a given subgrade layer in the Materials panel.]

The **Layer # Depth (Z in)** control sets the depth of the current subgrade layer.

The **Shoulder (X in)** and **Shoulder (Y in)** controls set how far the subgrade extends beyond the slabs in the **X** and **Y** directions. If the shoulder option is turned off, these controls will be inactive.

3.3 Dowels Panel

3.4

The **Dowels** panel sets the number and placement of the dowels connecting the slabs. This panel is only available if there are two or three slabs.

There are three options for specifying dowel positions, selected by the radio buttons: **Even Spacing**, **Wheelpath Spacing**, and **Manual Entry**. The first two options calculate all the dowel positions automatically; the third allows each dowel to be placed individually.

Under the first two options, the **Number of dowels** control sets the number of dowels. (Under **Wheelpath Spacing**, the number is restricted to be even.) It is not possible to change the number of dowels while in the **Manual Entry** option; the desired number of dowels should be set *before* that option is selected.

Under the **Even Spacing** option, the end dowels are set in from the edge of the slab by the distance specified in the **Offset from edge** control, and the remaining dowels are spaced evenly between them.

Under the **Wheelpath Spacing** option, the dowels are divided into two groups, each group centered around a theoretical "wheelpath." The cryptically named **WP CL @ +/-** control sets where the wheelpaths are - the title is an abbreviation of "Wheelpath centerline at +/- Y". If this value is set to 30, for instance, then each group of dowels will be centered around a line +/- 30 inches from the center of the slab. The dowels will be spaced apart by the distance set in the **Dowel Spacing** control.

The **Emb** control sets the embedment, or how far each dowel penetrates into the slab on each side.

The **Bonded Dowels** option intructs the mesher to fix the dowels laterally at every node. If this option is not selected, the dowels are assumed debonded along their entire length.

The **Gap (A)** and **Gap (B)** dimension controls - which refer to the dimensions shown in the diagram at the top of the panel - are used to set the dimensions for the gap around each dowel.

The numerical values resulting from the current dowel placement method will be shown in the spreadsheet in the bottom half of the panel. These values may not be edited unless the **Manual Entry** option is selected. When that option is selected, the **Number of Dowels** and related controls are inactivated, and it is possible to adjust all the dowel values individually. The user may specify the Y location, Z height (remember the coordinate system: Z is positive downwards, so the dowel Z coordinate will be a negative number), and the A and B values for each dowel individually.

Important Note: if either **Even Spacing** or **Wheelpath Spacing** is selected after the dowel values have been altered via **Manual Entry**, *all those changes will be lost*. Under either **Even** or **Wheelpath Spacing**, all the dowel values

are *always* calculated based solely on the parameters given. (The intention is that if the user needs to set dowel values manually, he or she will first use either **Even** or **Wheelpath** spacing to get the placement as close as possible to what is desired, and will then select **Manual Entry** and alter any values that require it.)

Note also that, in the case of a three-slab model, the dowel values affect the dowels at *both* joints simultaneously.

3.5 Materials Panel

3.6

The **Materials** panel sets the elastic material properties for the various components of the model.

There are four materials whose properties can be set: the slab(s), the subgrade(s), the dowels, and the dense liquid foundation. The dense liquid foundation layer lies underneath the bottom-most subgrade layer - or directly underneath the slabs, if there are no subgrade layers - and acts as a bed of springs underneath the nodes (i.e, a Winkler foundation).

For the slab and subgrade, the following properties can be set: **E**, **nu**, **alpha**, and **density**. For the dowels, the properties are: **E**, **nu**, **alpha**, and **diameter**. The dense liquid foundation has only a single property, the modulus of subgrade reaction **K**.

The dense liquid foundation also has the option, **No tension**, which means that the subgrade (or slabs) can "pull away" from the foundation.

For convenience, the drop-down list box for each material type contains a list of materials ("Soil," for instance) with pre-set values for the various material properties. Selecting a material from the list automatically sets the properties to those values. In the distribution, there is only one option listed for each area of the problem; to find out how to add new materials to the list, see <u>Adding New Default Materials</u>.

The **Subgrade** panel sets the properties for the currently selected subgrade layer; to select a different subgrade layer, position the mouse pointer on that layer in the elevation view and click the left mouse button (as in the <u>Geometry</u> panel.)

3.6.1 Adding New Default Materials

In the *scripts* directory under the root **EverFE** directory, there is a file called *default.mat*. As distributed, it looks like this:

```
SLAB
Concrete
4000 0.25 6e-6 8.7e-5
BASE
Soil
15 0.2 6e-6 0.00
FOUNDATION
Soil Foundation
0.150
DOWEL
```

```
Steel
29000 0.3 6e-6 1.0
```

The format should be fairly self-explanatory: the section under SLAB contains the list of possible slab materials, with their material properties in the order **E**, **nu**, **alpha**, **density**. The section under BASE lists possible subgrade materials, and so on.

To add a new material, simply insert its name and values into the file. For instance, suppose that a road is going to be constructed out of the rare material, *unobtanium*. We would insert the lines

```
Unobtanium
5000 0.10 6e-6 0.10
```

into the file in the SLAB section. Note that whichever set of properties appears first in the list will be the default selection when **EverFE** starts up.

3.7 Loading Panel

3.8

The **Loading** panel allows placement of loads on the road surface.

There are six different types of loads that can be placed: point, circular patch, and rectangular patch; and either single or axle placement. The **Load Type** control selects one of these load types; and the small graphic display shows the load and which dimensions are affected by which controls. For instance, an "Axle load - circular" requires specification of the radius, distance between the loads, and the load magnitude. (Note that, in the case of axle loads, the magnitude is the *total* magnitude; each of the two individual loads is alloted half the total.)
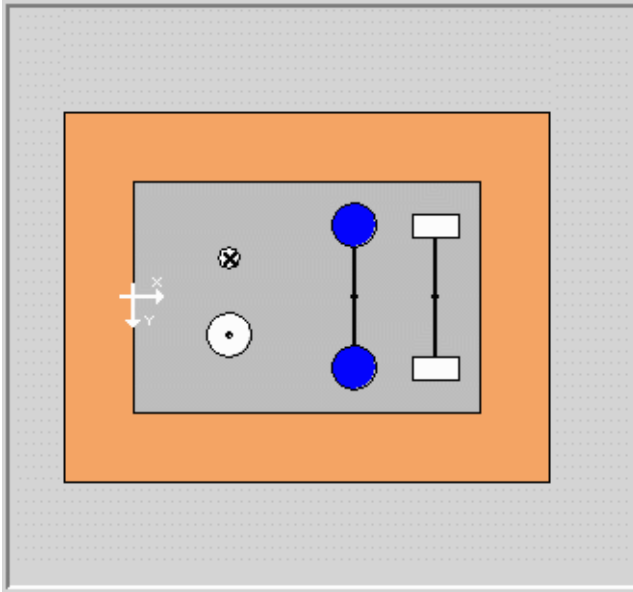
Important note: do **not** place loads over the gap between two slabs, or hanging over the edge of a slab. This will cause the solver to fail, or to give incorrect results.

Once a load type has been selected, press the **Place Load** button and move the mouse pointer to the plan view window. The **X** and **Y** values in the control panel track the mouse pointer; clicking the left mouse button places the load.

The currently selected load is colored blue; all other loads are colored white. The **X**, **Y**, and load-specific controls can be used to update the load parameters; and pressing **Delete Load** will remove the current load.

The **Delta T** controls can be used to specify a linear temperature gradient through the thickness of the slabs.

Plan view showing several loads placed. The current load is the one colored blue.
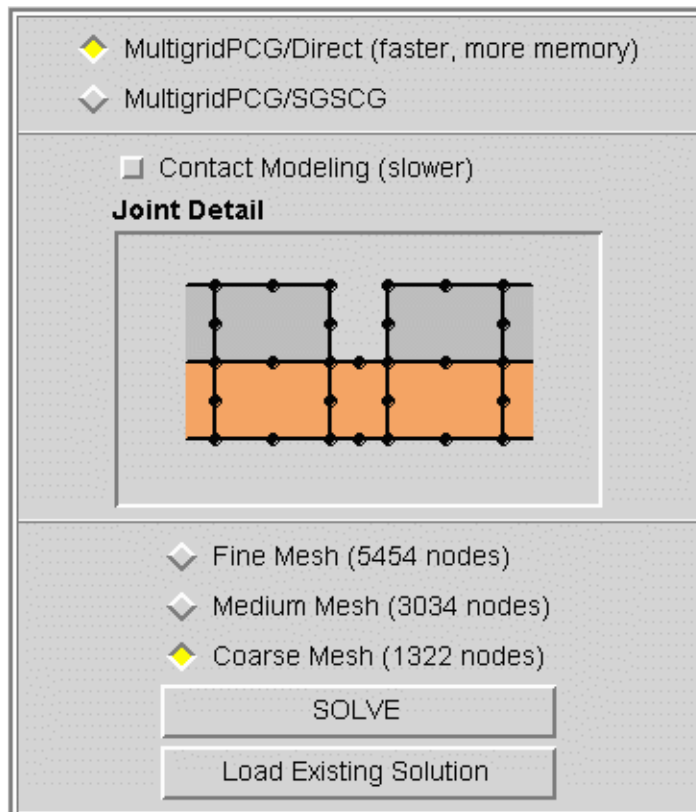
3.9 Meshing Panel

3.10

The **Meshing** panel sets the mesh parameters and invokes the solver.

The **MultigridPCG/Direct** and **MultigridPCG/SGSCG** controls tell the solver whether to use a direct solver or an iterative solver at the coarse level in the multigrid scheme. The direct solver is significantly faster, but requires more memory.

The **Contact Modeling** option affects how joints and the contact between the slab and the subgrade is meshed. In *contact modeling*, the bottom of the slab and the top of the subgrade are meshed as separate layers, which are internally constrained. This allows the slab to "pull away" from the subgrade under the effects of loading, and is in general more accurate. However, contact modeling requires several iterations of the solver to complete, and thus takes longer to solve.

If contact modeling is not selected, the bottom of the slab and the top of the subgrade are meshed as a single layer, and therefore deform together. This is less accurate, but faster to solve.

The **Fine**, **Medium**, and **Coarse** mesh radio buttons select the fineness of the mesh. A finer mesh means a more accurate solution, but a longer solution time. The node count for each mesh grade is given as a guide.
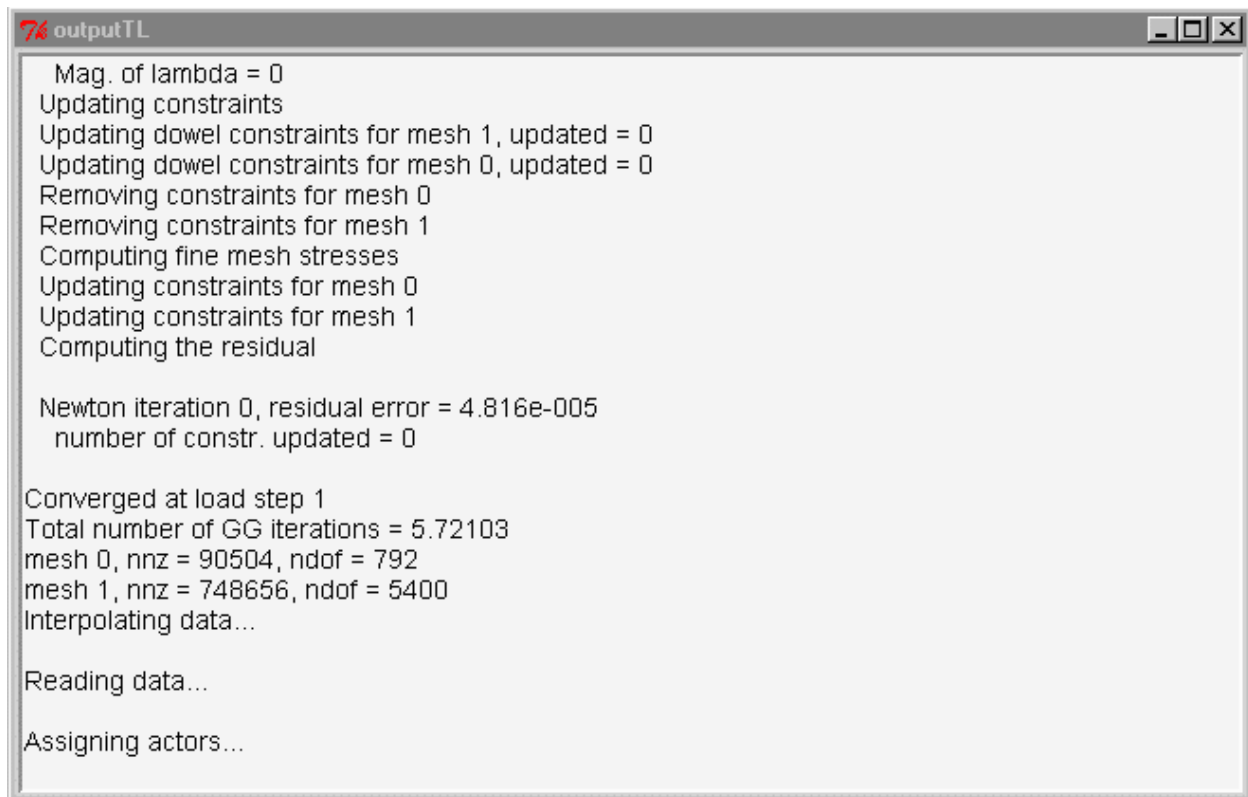
The **Load Existing Solution** button reads in the data files for an existing finite element solution, if one exists for this problem. It is not necessary to do this for a problem that has just been solved in the same session.

Before the solution - if one exists - is loaded, the user is warned that loading the solution will also load the geometry that existed at the time the solver was run. If the geometry has been changed subsequent to the FE solution, then the current geometry will be overwritten. This is necessary to ensure that the data being visualized corresponds to the geometry that **EverFE** believes to be current.

Finally, the **Solve** button invokes the finite element solver.

Depending on the processor speed, memory of your computer, and level of mesh refinement, this may take from a few minutes to several hours. If you expect a long solution time, we strongly recommend that you exit any other applications and deactivate any screen savers before starting the solver.

```
Mag. of lambda = 0
Updating constraints
Updating dowel constraints for mesh 1, updated = 0
Updating dowel constraints for mesh 0, updated = 0
Removing constraints for mesh 0
Removing constraints for mesh 1
Computing fine mesh stresses
Updating constraints for mesh 0
Updating constraints for mesh 1
Computing the residual

Newton iteration 0, residual error = 4.816e-005
  number of constr. updated = 0

Converged at load step 1
Total number of GG iterations = 5.72103
mesh 0, nnz = 90504, ndof = 792
mesh 1, nnz = 748656, ndof = 5400
Interpolating data...

Reading data...

Assigning actors...
```

The finite element solver

The finite element solver output is displayed in the solver window, shown above, while the solver is running. Assuming a normal run, when the solver completes, it will so indicate, and, after a few more progress updates, the solver window will disappear. At this point, the solution data has been written to disk in several files, and the results have been loaded into **EverFE's** data structures. You can view the data immediately, by going to the Deflections or Stresses panels, or you can reload the solution at a later time.

3.11 Aggregate Interlock Panel

The **Aggregate Interlock** panel provides controls for aggregate interlock modeling.

Aggregate Interlock provides shear transfer across the vertical joint between two slabs. Aggregate interlock can be modeled with either a simple linear model -- essentially modeling the joint as a spring with constant K -- or with a more complex non-linear model.

The **Aggregate Interlock** checkbox turns aggregate interlock modeling on and off. Since aggregate interlock modeling is meaningless unless there is a joint, this control is disabled unless there are two or more slabs.

The **Linear Model** and **Non-linear Model** radio buttons select which model will be used.

Under the linear model option, the **K (KIPS/in^3)** control sets the value of K, which may be interpreted as a spring stiffness per area of joint surface.

The non-linear model is more complex; for a given set of parameters – set in the controls -- an external program must be run to create the aggregate interlock model. Also note that the value of the gap between slabs (set in Geometry panel) has a large effect when using the non-linear aggregate interlock model.

The **Model Name** control selects from pre-generated models; as each one is selected, the parameters that generated it are displayed. **EverFE** is distributed with three pre-generated aggregate interlock models: *soft*, *medium*, and *hard*; with values of SigmaPU of 5000, 6000, and 7000 PSI (or 34.5, 41.4, and 48.3 MP in metric units) respectively.

To create a new model, select NEW MODEL from the **Model Name** control. This enables the controls for the user-adjustable parameters. When the parameters have been set, press the **Create Model** button; the program will prompt for the name of the new model, the external program will run -- displaying its output in a window the same way the solver does -- and when it is finished, the new model will be added to the list of those available.

3.12 Deflections Panel

The **Deflections** panel displays the deformation data of an existing solution.

The deformation data for each slab and the subgrade layer can be viewed independently, or in any combination. It is also possible to view the undeformed shape as well, for contrast. Deformed data is displayed as a green wireframe; undeformed data is displayed as a dark grey wireframe.

Use the checkboxes to select which sets of data you wish to view, and then press the **View Geometry** button. A new window will appear - if it isn't already displayed - and after a moment the requested figure(s) will appear.

The **Deformation Scale Factor** control sets the factor by which the actual deformation is magnified for the display.

Clicking the *Left* mouse button in the visualization window will rotate the display towards the position of the mouse pointer. Clicking the *Right* mouse button in the upper half of the screen will cause the view to zoom in; clicking in the lower half of the screen will cause it to zoom out. The *Middle* mouse button, if you have one, should cause the view to pan in the direction of the mouse pointer. Pressing the **Reset View** button on the panel will reorient the view to its initial setting.

View Deformed Geometry
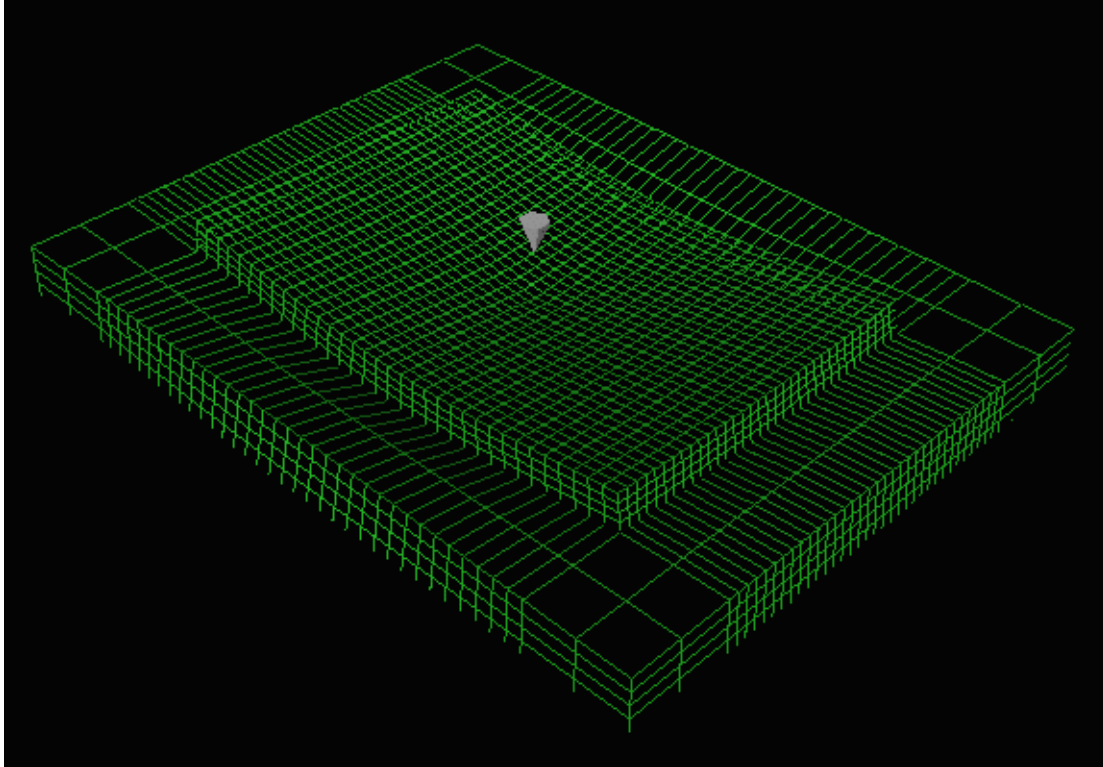
Slab 1   ☐ Deformed   ☐ Undeformed

Slab 2   ☐ Deformed   ☐ Undeformed

Slab 3   ☐ Deformed   ☐ Undeformed

Subgrade ☐ Deformed   ☐ Undeformed

Deformation Scale Factor
1000

View Geometry   Reset View

A deformed geometry visualization

3.13 Stresses Panel

The **Stresses** panel displays the stress data from an existing solution.

There are two primary modes for viewing stress data: as numerical data for a particular point; or in a stress colormap of a "slice" through the data.

EverFE uses the sign convention that tensile stress is positive, and compressive stress is negative. At each point, the three components of the principal stress are calculated, and sorted from *maximum* - most positive - *principal stress* to *minimum* - most negative - *principal stress*.

When a solution is loaded, **EverFE** searches the dataset to find the nodes with the maximum and minimum principal stress values for each slab. It is also possible to get data for any arbitrary point - **EverFE** will interpolate between the nodal data points.

To view numerical data, select the **Get numerical data from point** radio button. Assuming there is a current solution set loaded, the **X =/Y =/Z =** controls will activate, and a set of green crosshairs will graphically show the location of the currently selected point.

Once the desired point is selected, pressing the **Get Data** button will search and interpolate for the point. (It is also possible to select a point from the graphic stress display - see below.)

The data is displayed in a separate window, which can be called up or hidden by the **Show/Hide Data Window** button. There are two parts to the data window: the **Max/min principal stress values per domain** display, and the **Data for selected point** display.

The **Max/min principal stress values per domain** table lists the maximum and minimum nodal principal stress values for each slab, and the corresponding nodal coordinates. Note that it is possible that more than one node could have the same principal stress value; in this case, only one set of coordinates will be displayed.

In the case of the display shown here, we can see that in Slab 1, the maximum nodal principal stress value is 55.088 PSI, and this occurred at the point (114.641, 24.0, 0.0). Similarly, the minimum nodal principal stress value was -117.919, at point (112.823, -24.0, -9.0).

The **Data for selected point** display provides a great deal of data about a given point. The table displays:

X	Y	Z
29.0 in	0.0 in	-3.0 in
DX	DY	DZ
0.000352 in	0.000447 in	0.007046 in
SigmaXX	SigmaYY	SigmaZZ
-1.09228 PSI	1.15008 PSI	-0.515611 PSI
TauXY	TauYZ	TauZX
0.237389 PSI	-0.025389 PSI	0.177528 PSI
Max PS	Med PS	Min PS
1.17497 PSI	-0.466295 PSI	-1.16648 PSI

- The coordinates - **X**, **Y**, **Z**
- The displacement vector - **DX**, **DY**, **DZ**
- The components of the stress tensor - **SigmaXX**, **SigmaYY**, **SigmaZZ**, **TauXY**, **TauYZ**, **TauZX**
- The principal stress values, sorted from maximum to minimum - **Max PS**, **Med PS**, **Min PS**

To display a colormap of the stress in a selected plane, select the **View colormapped stress plane** radio button. **EverFE** displays stresses by mapping either the *maximum principal stress* at each point, or the *minimum principal stress* at each point, to a color. Since tensile stress is positive and compressive stress is negative, mapping the maximum principal stress shows the *most tensile* stress at a given point, while mapping the minimum principal stress shows the *most compressive* stress. Note that it is possible for the maximum principal stress at a given point to be compressive, or for the minimum principal stress to be tensile. The **Max TENSILE** and **Max COMPRESSIVE** radio buttons set whether the stress being mapped is the maximum principal or the minimum principal.

Since data can only be viewed in slices, you must select which plane you want the slice to be in - **Y-Z**, **X-Z**, or **X-Y** - and what slice of the data you wish to view.

The coordinates - **X**, **Y**, and **Z** - indicate where the slice will be cut. Data can only be viewed along the planes of the mesh; clicking the coordinates up or down will automatically increment them to the next mesh plane. To assist the user in selecting planes, a red line is drawn in the geometry display, indicating where the plane will be cut.

The **Color map scaling** options govern whether the colormap for the plane of data being viewed is scaled by the global maximum and minimum values of stress, or by the maximum and minimum for that plane only. In either case, a color map scale with numerical indices is displayed in the visualization window.

The color mapping is always scaled with blue being negative - i.e, compressive - and red being positive - i.e, tensile. Green is always neutral. This may mean that the color map does not reach all the way to the maximum red color, or the maximum blue color; this is necessary, to ensure that blue colors always indicate negative values and red colors always indicate positive.

Checking the **View direction** box will display lines radiating from the colormap in the direction of maximum principal stress. The **Scale Factor** value is used to scale these lines.

When all the desired options have been set, pressing the **View Stress** button will bring up the stress display. It may take a few moments, especially with large datasets, or for the initial view.

Clicking the *Left* mouse button in the visualization window will rotate the display towards the position of the mouse pointer. Clicking the *Right* mouse button in the upper half of the screen will cause the view to zoom in; clicking in the lower half of the screen will cause it to zoom out. The *Middle* mouse button, if you have one, should cause the view to pan in the direction of the mouse pointer. Pressing the **Reset View** button on the panel will reorient the view to its initial setting.

Pressing **P** while the mouse pointer is over the stress colormap will load the coordinates for that point into the **Get numerical data from point** controls, and search for the data from that point.
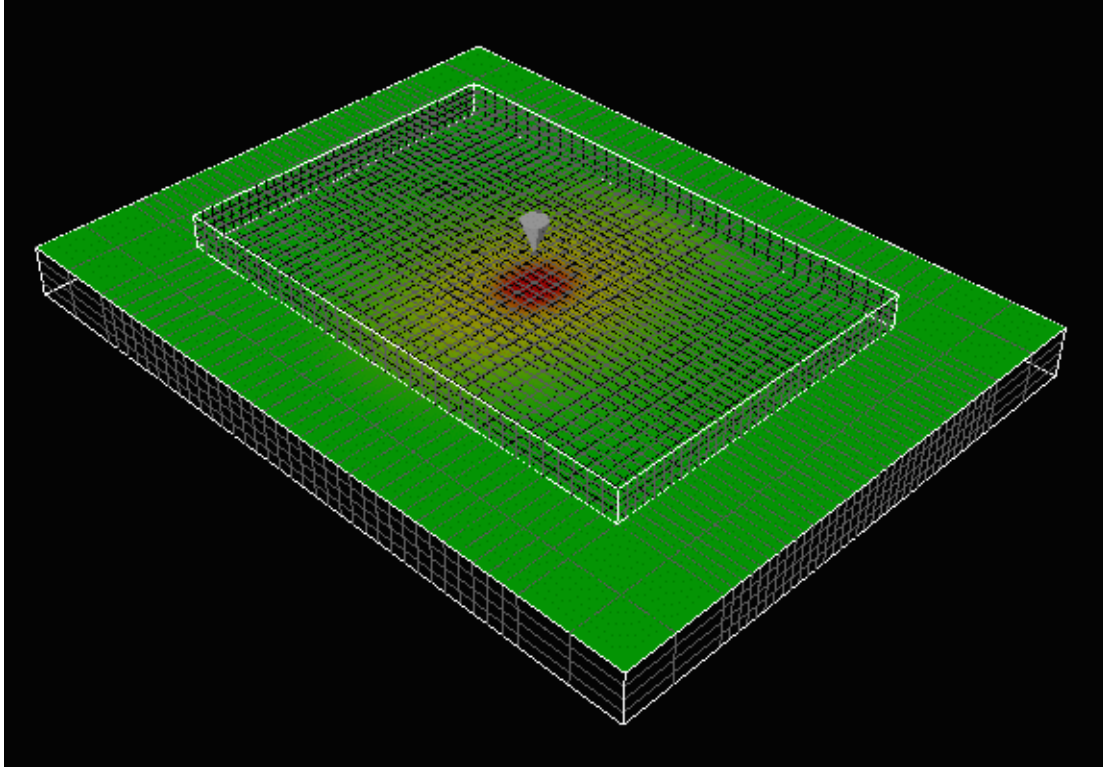
A stress visualization

4. Tutorial

The purpose of this tutorial is to guide the user step-by-step through a simple EverFE problem, from geometry definition to viewing results. For detailed instructions on various aspects of EverFE, consult the documentation pages for those sections.

This is the problem that we will solve in this tutorial (note: this project is included in the EverFE distribution as "test2.prj"):
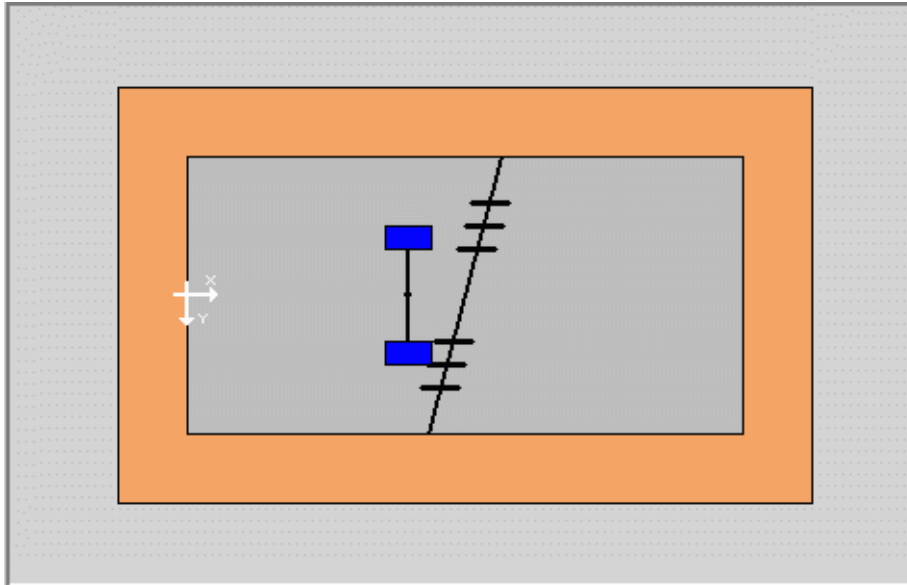

Plan view of tutorial problem
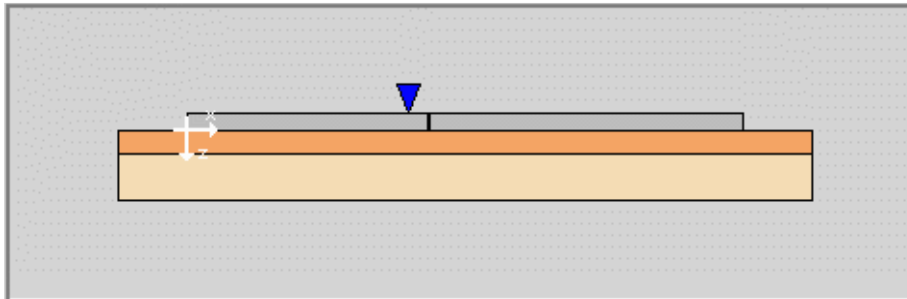
Elevation view of tutorial problem

Step 1 - Start EverFE

Start EverFE by double-clicking on the "EverFE" icon on the EverFE folder; or by selecting "EverFE" from the Start menu or the desktop (if it has been added to either of those locations.) After a few moments, the main EverFE window will appear.

The default system of units is English, which will be used for this model problem. To develop a model using metric units, go to the File menu and select New; a dialog box will appear prompting you to select either English ("U.S. Customary Units") or metric. Note that once a unit system is chosen, it cannot be changed for that project.
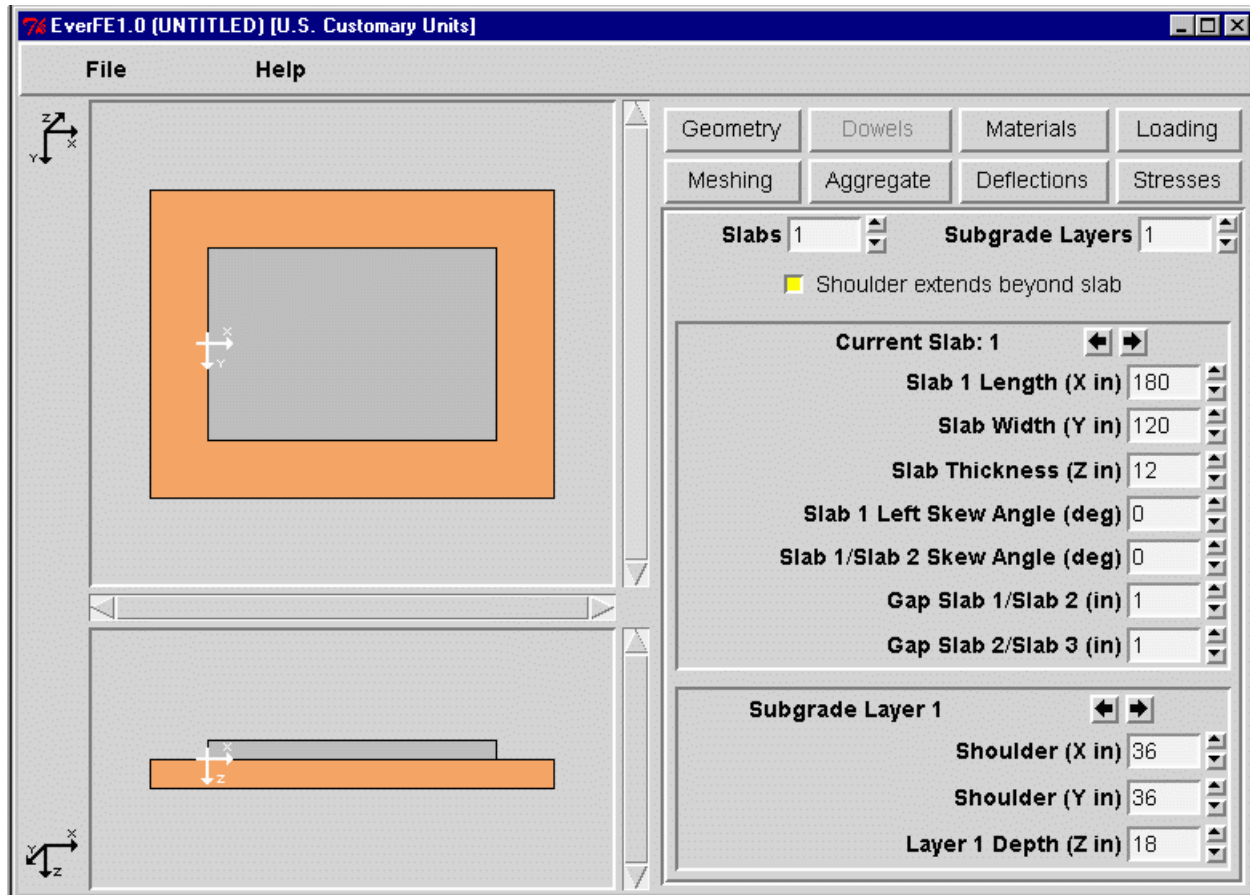
The main EverFE window

Step 2 - Define Geometry

The next step is to define the problem geometry. The "Geometry" panel is displayed when EverFE starts up; it can be displayed by pressing the Geometry button on the button panel at any time. To create the geometry for the tutorial problem, do the following:

1) Set the "Slabs" control to "2"
2) Set the "Subgrades" control to "2"
3) Make sure the "Shoulder extends beyond slab" checkbox is checked
4) Set "Slab 1 Length (X in)" to "144"
5) Set "Slab Width (Y in)" to "144"
6) Set "Slab Thickness (Z in)" to "9"
7) Set "Slab 1/Slab 2 Skew Angle (deg)" to "15"

   Now click on the right-arrow button to the right of "Current Slab: 1" The label changes to read "Current Slab: 2" and the dimension controls for the second slab appear.

8) Set "Slab 2 Length (X in)" to "144"

   In the bottom part of the panel, where it says "Subgrade Layer 1":

9) Set "Shoulder (X in)" to "36"
10) Set "Shoulder (Y in)" to "36"
11) Set "Layer 1 Depth (Z in)" to "12"

12) Click the mouse on the second (bottom-most) subgrade layer in the elevation view, or click the right-arrow button on the subgrade panel; notice that the label for the bottom part of the panel changes to "Subgrade Layer 2"

13) Set "Layer 2 Depth (Z in)" to "24"

At this point, your plan and elevation displays should look like this:
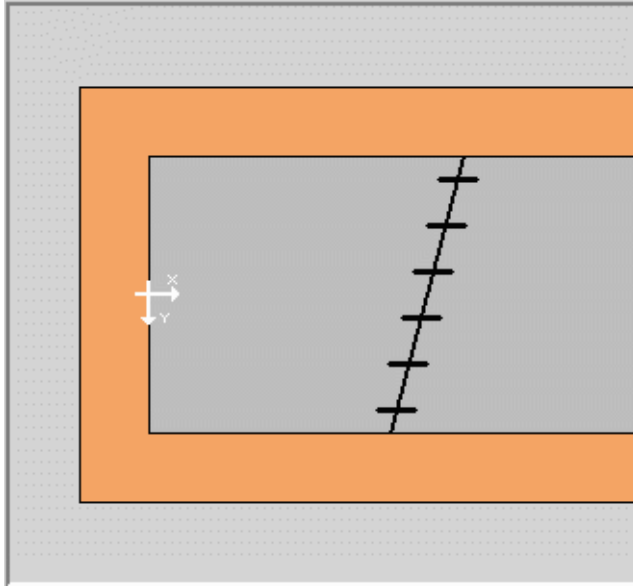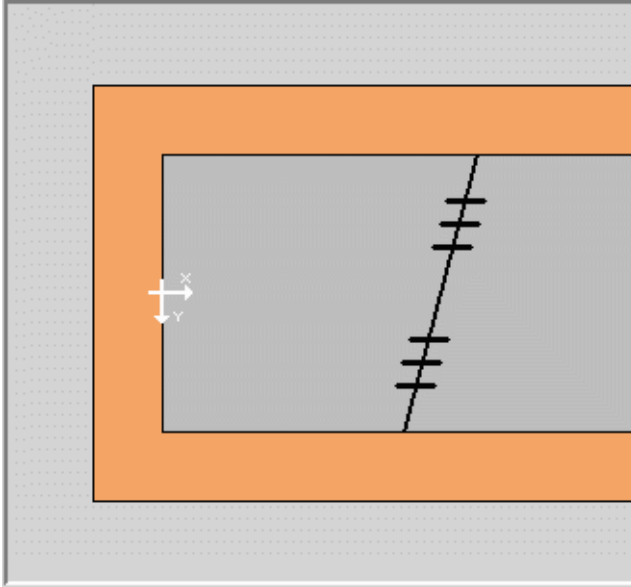

Plan view after Step 2
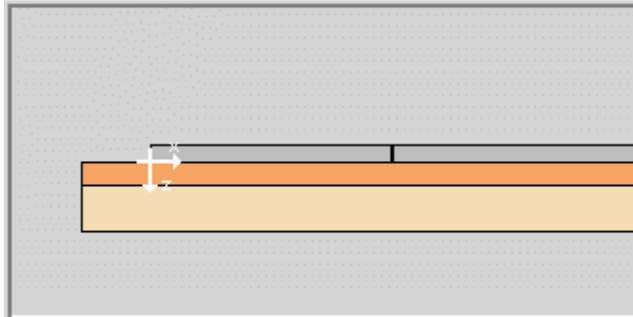
Elevation view after Step 2

Step 3 - Place Dowels

Press the "Dowels" button on the button panel; the dowel controls will appear. To create the proper dowel spacing for the tutorial project:

1)   Select the "Wheelpath Spacing" option
2)   Set "Number of dowels" to "6"
3)   Set "WP CL @ +/-" (the wheelpath centerline) to "36"
4)   Set "Dowel Spacing" to "12"
5)   Set "Embed" to "9"

At this point, your plan display should look like this:

Plan view after Step 3

Step 4 - Set Material Properties

Press the "Materials" button on the button panel; the material properties controls will appear. To set the material properties for the tutorial project:

1) Click on the first (upper-most) subgrade layer in the elevation view
2) Adjust the "E" constant for "Subgrade #1" to "10.0"
3) Click on the second (bottom-most) subgrade layer in the elevation view
4) Adjust the "E" constant for "Subgrade #2" to "50.0"

Step 5 - Place Loads

Press the "Loading" button on the button panel; the loading controls will appear. To place the load for the tutorial project:

1) From the "Load Type" list box, select "Axle load - rectangular"
2) Press the "Place Load" button
3) Move the mouse pointer to the plan view. Note how the "X" and "Y" values on the control panel track the mouse pointer. Position the pointer in the vicinity of X = 115, Y = 0, and then click the left mouse button. An axle load appears.
4) If you didn't get it placed at exactly X = 115, Y = 0, then adjust the X and Y controls to those values. The load will move as you modify the X and Y values.
5) Set "A" to "24," "B" to "12," "D" to "60," and "Mag" to "22." Watch how the load changes as you alter the dimensions.

At this point, your plan display should look like this:
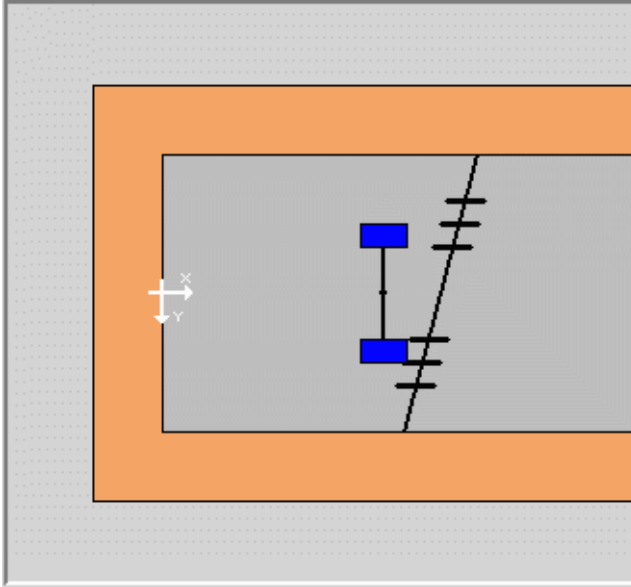
Plan view after Step 5

Step 6 - Set Aggregate Interlock

For this problem, we'll turn on one of the optional features – aggregate interlock. Press the "Aggregate" button on the button panel. To turn on aggregate interlock:

1)   Select the checkbox labeled "Aggregate Interlock"
2)   Under the "Linear Model" option, adjust the value of K to 2.0

At this point, all the relevant parameters have been entered, and we're ready to solve the problem. First, we should save it:
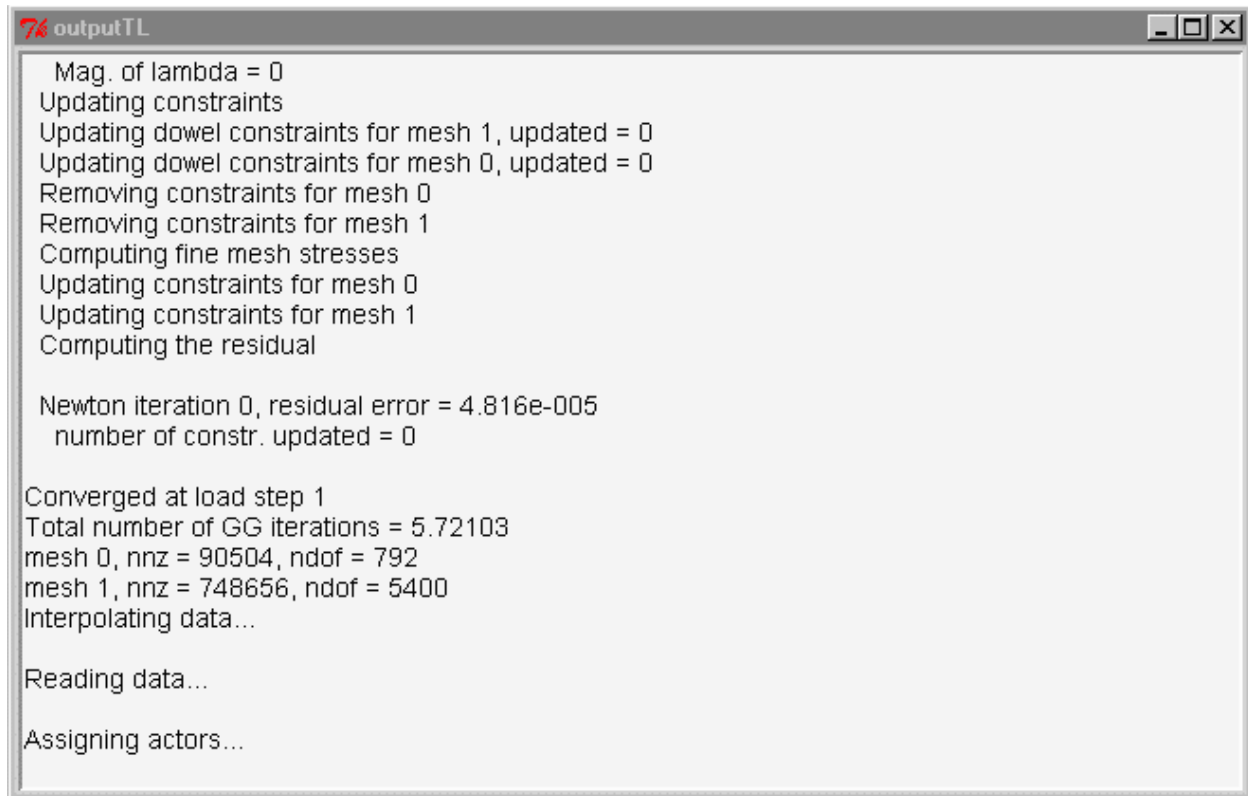
1)   Select "Save" (or "Save As") from the "File" menu
2)   Enter a name for your project, such as "tutor.prj"
3)   Press the "OK" button

Step 7 - Solving

Press the "Meshing" button on the button panel; the meshing controls will appear. To solve the tutorial problem:

1)   Select the "MultigridPCG/Direct" option - this will cause the solver to run faster, but requires more memory
2)   Make sure that the "Contact Modeling" option is turned off
3)   Select the "Coarse Mesh" option
4)   Press the "SOLVE" button
5)   A message appears, essentially asking if you're sure you want to run the solver now. Press "OK"

6) The solver will run for a few minutes - this would be a good time to go and get a cup of coffee. While it's running, it's output will be displayed in this window:

7) When the output window disappears, the solution is complete

Now that the solution has been run, the data is written to disk in several files stored under the project name; there is no need to "save" the solution data. The results can be viewed either now, or at a later time. If you want to view the results of an already-solved problem, see the explanation of the "Load Existing Solution" function in the Meshing panel documention.

Step 8 - Viewing Displacement Data

Press the "Deflections" button on the button panel; the deformation viewing controls will appear. To view the displacement data:

1) Select the "Slab 1 - Deformed," "Slab 2 - Deformed" and "Subgrade - Deformed" options

2) Press the "View Geometry" button. After a few moments, the data visualization window will appear, which looks like this:
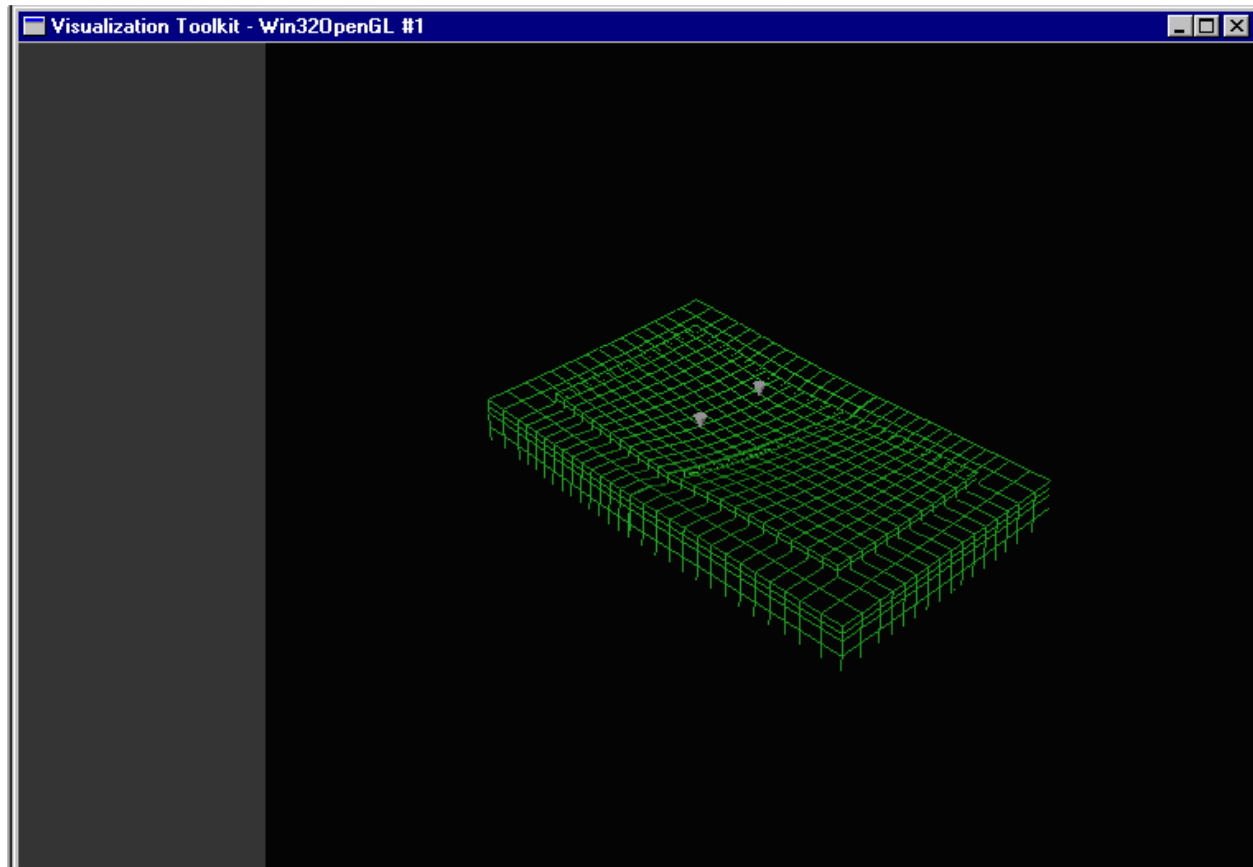
3) Click the right mouse button a few times in the upper half of the screen to zoom in on the image, and you will see something like this:
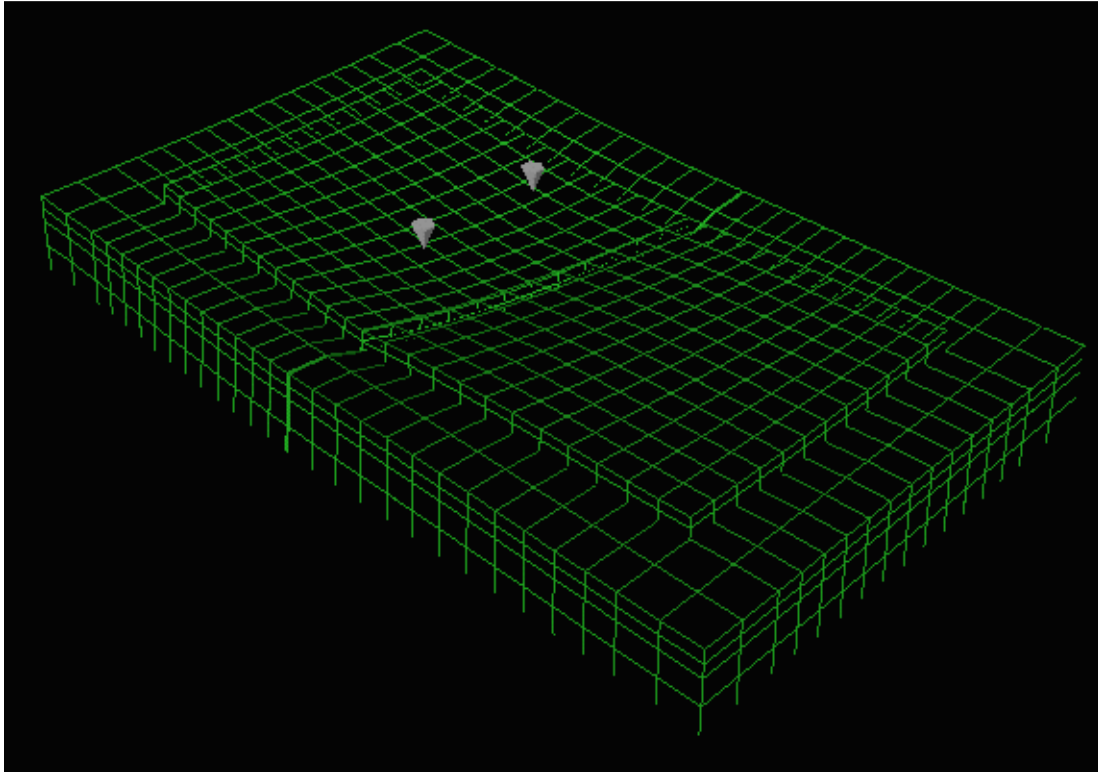

The small grey cones indicate the positions of the loads. The deformed shape of the slabs and the subgrades is shown by the green wireframe - naturally, the magnitude of the deformation is greatly exaggerated; the scaling factor can be controlled by the "Deformation Scale Factor" control. For more details on viewing deformed images, see the documention on the Deformed control panel.

Step 9 - Viewing Stress Data

Press the "Stresses" button on the button panel; the stress viewing controls will appear. To view the stress data:

1) Select the "View colormapped stress plane" radio button.
2) Select the "X-Y" plane to view
3) Press the down arrow on the "at Z (in) =" control until the red line on the elevation view lines up with the top surface of the slab. This will give us a "slice" of the very upper surface of the slab.

4) Press the "View Stress" button. The visualization window will now look like this:
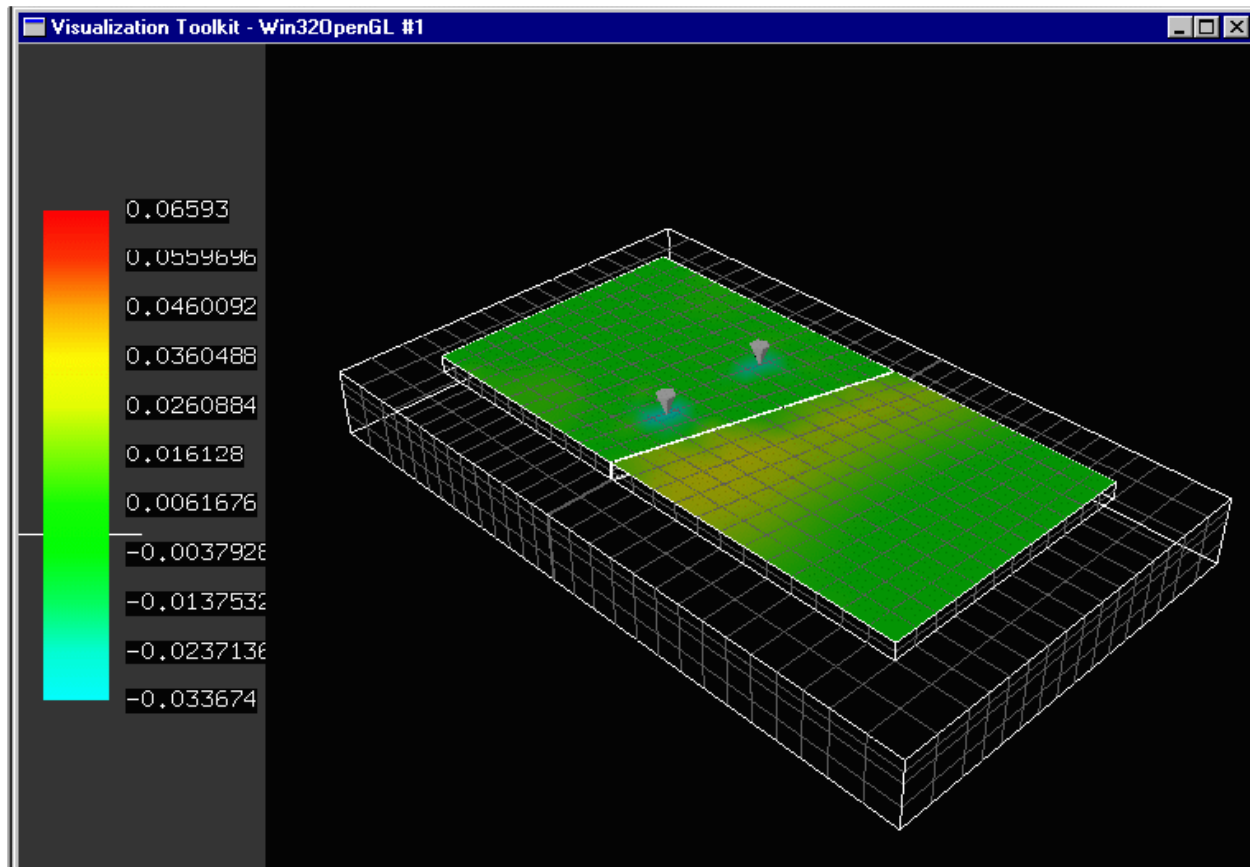
5) The color map for this slice is being scaled by the global maximum and minimum stress values, which is why the colors are all so close to green (representing neutral stress.) However, this fact allows us to instantly read off the maximum and minimum global values for the maximum tensile principal stress; those values are 0.06593 and -0.033674, respectively.

6) To make this particular slice of data more interesting, select the "Color map scaling: Local" option, and press "View Stress" again. The image now looks like this:
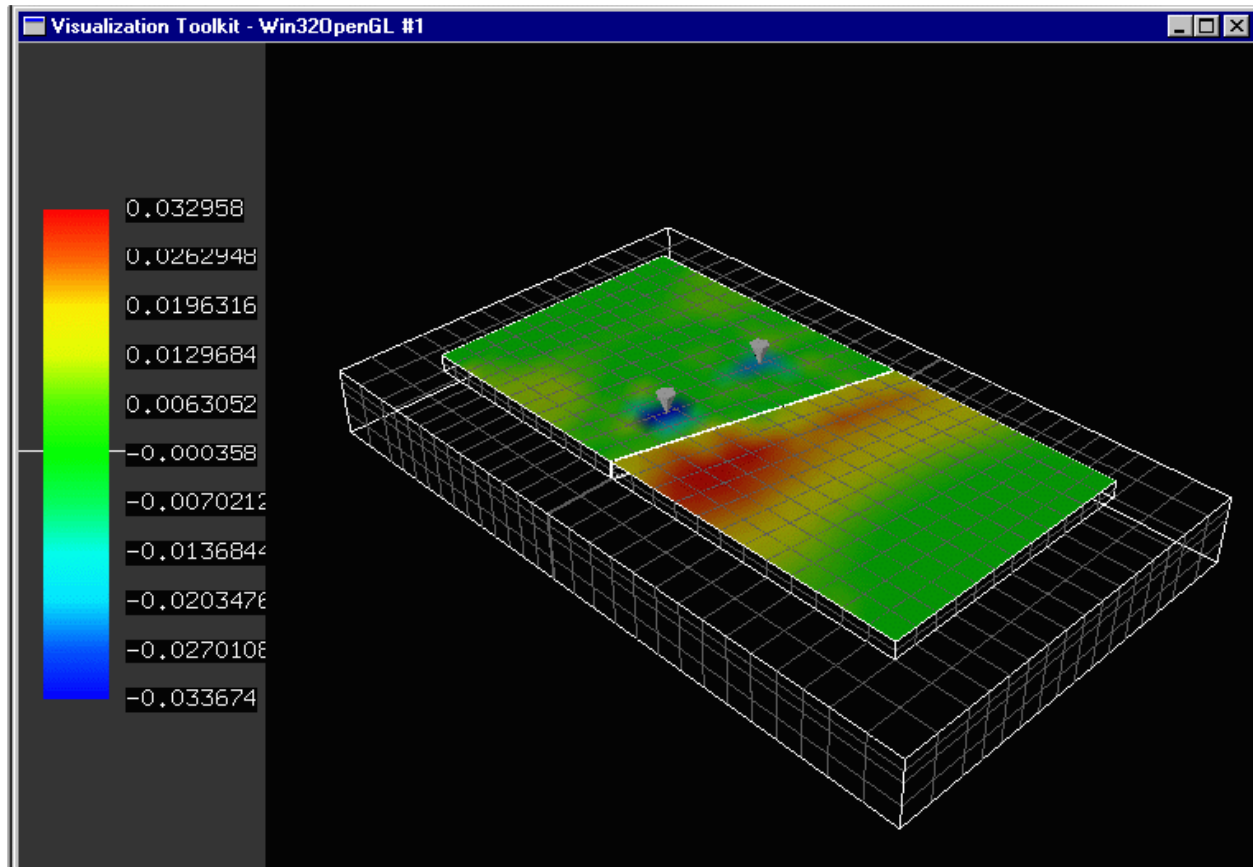
7) For more details on viewing stress data, see the documention on the Stressed control panel.

At this point, you should experiment with viewing the data from this problem, or from the two example projects included in the distribution. The documentation pages for the various control panels go into much more detail on the uses of the various aspects of **EverFE**; you should familiarize yourself with them.

This concludes the tutorial.

5. List of Known EverFE Bugs

The following is a list of bugs (or just oddities) of **EverFE** that are known at this time.

- **Start-up under Windows 95** - this isn't really a bug; but when EverFE is started under Windows 95 - at least on our development PC - it takes a fairly long time before the EverFE window appears; time in which it's easy to believe that nothing has happened. Be patient; the application really is starting up.

- **Control panels garbled after minimizing** - if the EverFE window is minimized, switching between control panels afterwards is likely to cause them to become garbled - as though several panels were being overlaid at once. Switching back and forth between the panels several times usually clears this up.

- **Load placement in illegal areas** - the solver can only deal with loads that are placed, in their entirety, on a slab surface; but it is possible to place loads in other places - for instance, by placing a load so that part of it hangs over the edge of a slab, or by changing the geometry underneath the loads. In this case, the solver will either crash, or - worse - give incorrect results.

- **Subgrade layer controls not updating properly** - occasionally, clicking on a subgrade layer does not bring up the correct controls. Saving, quitting, and restarting EverFE appears to fix this.

- **Closing VTK window exits prematurely** - if the VTK window - the graphic window that displays the displacement and stress results - is closed by clicking the windows "X" button in the upper right corner, then EverFE will exit immediately, without prompting for saving any changes.

- **Stress colormap not initially displayed** - when the stress display is first called up, the colormap may not be displayed. Clicking a mouse button in the window, to either rotate or zoom the image, will cause it to be displayed properly.

- **Getting data from point on edge of data set** - sometimes, when attempting to get numerical stress data on a point by pressing "P" in the VTK window, the data all comes up as "N/A." This means the coordinates of the point, as obtained from the VTK window, were so close to the edge that the data interpolation routine decided the point was outside the data set. To fix this, simply adjust the appropriate coordinate by a small amount, until you get results.